Airflow and Dispersion Around Multiple Buildings

R. L Lee
S. T. Chan
J. M. Leone Jr.
D. E. Stevens

February 1, 1999

Lawrence Livermore National Laboratory

DISCLAIMER

Airflow and dispersion around multiple buildings

R. L. Lee, S. T. Chan, J. M. Leone, Jr., D. E. Stevens
Lawrence Livermore National Laboratory
University of California
Livermore, CA 94551
Email: lee34@llnl.gov

Abstract

A three dimensional, finite element-based, flow and dispersion model is used to simulate the transport and fate of hazardous releases in the atmosphere. Numerical results are presented for two experimental studies: (1) Airflow and dispersion over multiple blocks in a wind tunnel; and (2) Tracer study of a point release in the neighborhood of a building complex.

1 Introduction

Predictions of airflow and dispersion over urban areas continue to be a challenging problem because of the presence of extremely heterogeneous surface features within typical centers of population. For the case where atmospheric conditions are highly stable (low Froude number cases), the local wind tends to flow around structures rather than over them. Thus pollutants that are released at or near the ground tend to persist at relatively low levels and have the potential of inducing negative health effects on the population. We are developing numerical models to simulate the flow and dispersion of releases around multi-building complexes. These models will be used to assess the transport and fate of releases of hazardous agents within urban areas and to support emergency response activities.

There are already a number of models that have been developed to simulate flow and dispersion around urban settings. A recent collection of these papers can be found in the Proceedings of the 2nd International Symposium of Computational

Wind Engineering.[1] Most of the simulation studies presented in the literature are focused on the simple setting of single buildings and several of these model calculations have been compared with wind tunnel experiments[2, 3]. There is considerably fewer reported work in the open literature on the influence of multiple buildings, vegetation, surface heating and atmospheric stability on flow and dispersion.

An important focus of the LLNL building flow and dispersion effort is the development and valuation of numerical models to assess the transport and fate of chemical and biological releases within urban areas. We are collaborating with scientists at the Fluid Modeling Facility to conduct laboratory experiments at the NOAA/EPA wind tunnel. These experiments, which are focused on simulating flow and dispersion of releases within multiple block arrangements, are intended to provide datasets to validate our models. In addition to wind tunnel experiments, we also have plans to conduct field experiments to support our model valuation effort. In this paper we will report on some initial wind tunnel simulations and the application of our building dispersion model on a field study involving releases around a three buildings.

2   The flow and dispersion model (FEM3CB)

FEM3CB is an adaptation of the FEM3C[4,5] model that was developed to simulate flow and dispersion of heavier-than-air gases. The current model is significantly more efficient than the earlier version and includes several new features that are specific to the building dispersion application. The model equations and a detailed description of this model can be found in Chan and Lee.[6] Some relevant aspects of the model are:

- Finite-element technique
- Fully 3-D
- Use of projection method and fully implicit time integration scheme
- Generalized anelastic formulation
- Explicit treatment of terrain and surface obstructions
- Option of k-ϵ and K-theory turbulence submodels
- Handles large density variations

FEMCB uses a modified finite element technique to provide great flexibility in the arbitrary grading of the mesh in order to economize on computational costs. While the standard Galerkin finite element formulation can be easily applied to very general meshes, it is also computationally expensive to use. In FEM3CB we employ a number of simplifications to the traditional finite element method to permit the code to be computationally affordable for three-dimensional problems.

Rather than solving the coupled set of momentum and continuity equations, the model uses a segregated approach in which the system is decoupled by using a consistent pressure Poisson equation in lieu of the continuity equation. This projection method was first proposed and analyzed by Chorin[7] and many variants of this algorithm have been used in the computation of incompressible flows. Some key features that contribute to the computational efficiency of the model include:

- Logical (i, j, k) but non-uniform grid system
- Highly efficient conjugate gradient pressure solver
- Fully implicit time integration treatment for diffusion

The K-Theory turbulence model in FEMCB employs the widely used similarity theory and formulae for the atmospheric boundary layer. This formulation is generally appropriate for building dispersion applications since the spatial scales of interest are typically within the surface boundary layer (a few hundred meters above ground) and over relatively short time scales. Using this formulation, the vertical profiles of velocity and temperature (departure from a reference value) are given by:

$$u = u_* / \kappa \, [\, \ln \{ y/y_o \} + \psi \,] \, ,$$

and

$$\theta = \theta_* / \kappa \, [\, \ln \{ y/y_o \} + \psi_\theta \,] \, .$$

In the above equations, u_* is the friction velocity, y_o is the ground surface roughness scale, θ_* is the friction temperature, κ is the von Karman constant (=0.41), and ψ, ψ_θ are the similarity functions used by Haroutunian[8] to account for atmospheric stability.

The boundary conditions used in the model are typically a combination of Dirichlet (specified velocity and temperature) or specified normal and tangential stresses and heat flux. At the outflow, "natural" (zero stress and heat flux) boundary conditions appear to work well except for the most difficult simulations.

Pollutant sources are modeled as either as a puff (limited time duration) or as continuous releases. In the Euler representation, point sources are distributed over a few, highly graded, mesh cells with the spatially integrated pollutant mass flux equal to the required source strength. Since the pollutants of current interest are assumed to be inert, concentration calculations can be performed from a velocity field that has been computed ahead of time.

3. Numerical simulations

Numerical results are present for two simulations: (a) Flow and dispersion for a multi-building array within a wind tunnel; (b) Downwind dispersion of a tracer release near a 3-building complex. The first simulation is an initial attempt to validate FEMCB with wind tunnel experiments at EPA. The goal of the second simulation is to compare the model results with a field experiment that has been conducted at a multi-building complex. These simulation studies are currently ongoing and only some initial results are presented at this conference.

3.1 Simulation of flow and dispersion over a multi-building array within a wind tunnel

We are collaborating with researchers at the Fluid Modeling Facility to conduct laboratory experiments at the NOAA/EPA wind tunnel. These experiments are focused on the physical modeling of flow and dispersion of releases within multiple block arrangements. The initial experiments are based on two sets of building configurations, a 7-row, two-dimensional, street canyon configuration and a three-dimensional, 7-by-7 multi-block configuration. The data that will be available from these experiments will include flow visualization, mean wind and concentration profiles and turbulence intensity measurements. We will present some initial simulations from the 7-by-7 block configurations in this paper.

In the 7 x 7 multi-building experimental setup, the buildings are represented by cubical blocks of 15 cm^3 in size. The spacing between the blocks can be varied but the distances between blocks are fixed at 15cm in the initial experiments. Figure 1 shows the 2.95 m (ℓ) x 1.85 m (w) x 2.1 m (h) computational domain representing the experimental test section and the multi-block surface arrangement. The mean wind, based on the K-theory turbulence model used in the simulation, is assumed to be logarithmic at 3.0 m/s and the air is neutrally stable with $\psi = 0$. The continuous point source (source rate = 0.006 g/m^3) is located at the center plane, -0.5 m (upstream of the first row of blocks) and 0.025 m above the bottom plane of the wind tunnel. Only one-half of the problem was calculated due to symmetry.

Figures 1 and 2 show projections of the velocity field on a vertical (z = 0) and horizontal (y = 0.03 m) plane at t = 4 s which is approximately steady state. The horizontal view of the wind field depicts a relatively complex flow pattern with multiple weak recirculations within the gap regions and channel effects especially near the inlet plan of the block system. Figure 3 shows a contour plot on the y = 0.03 m plane at t = 5 s. The shape of the concentration pattern reflects the transport of the plume due to the channeling effects between the blocks in the flow direction. The data from this wind tunnel study is not yet available therefore no quantitative comparisons between experiments and model simulations can be made at this time.

3.2 Flow and dispersion around a multi-building complex

A series of field experiments were carried out at the Casaccia Nuclear Research Center (Italy) to study wake effects downwind of an airborne release around a building complex.[9] The three buildings downwind of the release are shown in fig. 4. The release point, which is marked with a "+" on the right hand bottom of this figure, is about 14 m above ground level. The height of building A is 6 m and buildings B and C are 16 m high. Two anemometers were used to measure wind speeds and directions at 10 m and 40 m. In addition, an extensive array of air samplers were deployed at various distances downwind to collect concentration data (see fig. 4).

The finite element mesh for the 3-building simulation is shown in fig. 5. Two experiments in which the atmospheric conditions were neutrally stable were selected for the calculations. The first experiment is based on a $250°$ wind direction at 5.2 m/s and a unit (1 g/s) release rate. The observed surface concentration pattern shown in fig. 6 depicts a fairly concentrated plume spreading laterally and downwind along the centerline. The simulated plume displayed in fig. 7 shows a very similar pattern with the lower concentration contours agreeing fairly well with observations but with the centerline high concentration regions reflecting much higher concentration values. This may be due to the stagnation region between the L-shaped building and the two downstream buildings. The agreement at low concentrations (see figs. 8 and 9) is also displayed for the second experiment with the wind direction at $205°$ at 3.7 m/s. Here, the concentration pattern and the shape of the low-level contours are fairly well reproduced, however the concentration levels close to the source is much higher in the numerical simulation. The persistence of high concentrations near the release is probably caused by deficiencies of the subgrid model. We are investigating the dependence of these results to different subgrid models.

References

1. Meroney, R. and Bienkiewic, B., (eds.), Proceedings of the 2nd International Symposium of Computational Wind Engineering, Computational Wind Engineering 2, *Journal of Wind Engineering & Industrial Aerodynamics*, **67** & **68**, 1997.

2. Cowan, I. R., Castro, I. P., Robins, A. G., Robins, Numerical considerations for simulations of flow and dispersion around buildings, *Journal of Wind Engineering & Industrial Aerodynamics*, **67** & **68**, pp. 535-545, 1997.

3. Li, Y. & Stathopoulos, T. Numerical evaluation of wind-induced dispersion of pollutants around a building, *Journal of Wind Engineering & Industrial Aerodynamics*, **67** & **68**, pp. 757-766, 1997.

4. Chan, S., FEM3C - An improved three-dimensional heavy-gas dispersion model: User's Manual, UCRL-MA-116567 Rev. 1, Lawrence Livermore National Laboratory, Livermore, CA., 1994.

5. Chan, S., Numerical simulations of LNG vapor dispersion from a fenced storage area, *Journal of Hazardous Material*, **30**, pp. 195-224, 1992.

6. Chan, S. and R. L. Lee, A model for simulating airflow and pollutant dispersion around buildings, UCRL-JC-132242, also in *Proceedings of Air Pollution 99*, July 27-29, 1999, San Francisco, CA, 1999.

7. Chorin, A. J., Numerical solution of the Navier-Stokes equations, *Mathematics of Computation*, **22**, pp. 745-762, 1968.

8. Haroutunian, V., A time-dependent finite element model for dispersion of gases heavier-than-air, Ph. D. dissertation, UMIST, Manchester, U. K., 1987.

9. Cagnetti, P., Downwind concentrations of an airborne tracer released in the neighborhood of a building, *Atmospheric. Environment*, **9**, pp. 739-747, 1975.
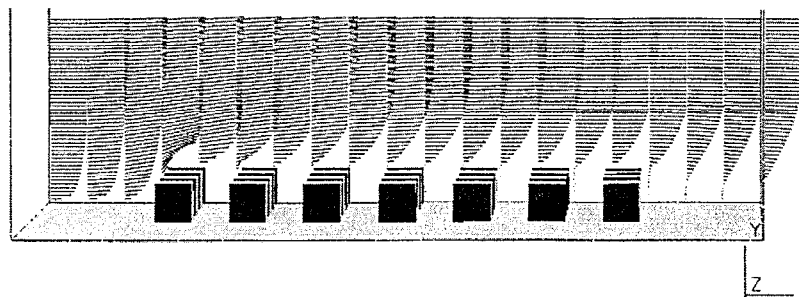
Acknowledgements

Figure 1: Simulated flow over the 7x 7 block configuration. Velocity profiles projected on the vertical plane of symmetry.
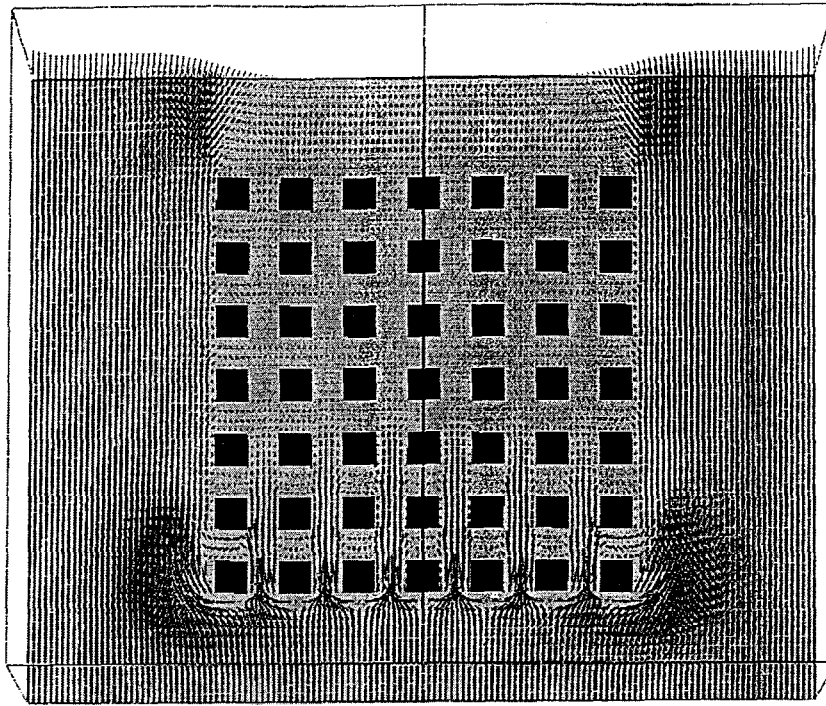
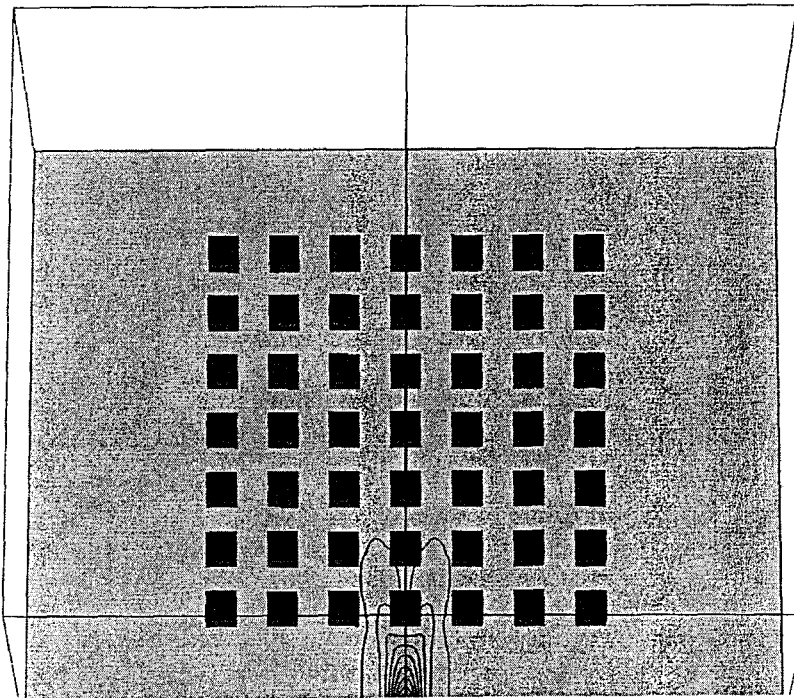Figure 2: Velocity field projected on a horizontal plane at y = 0.03 m above the bottom plane.

Figure 3: Concentration contour plot on the y = 0.03 m plane with a point source release of .006 g/s at (-.5, 0, .025) m. Contour levels are shown for 0.5 g/m^3 and 1.0 to 10.0 in 1 g/m^3 intervals.
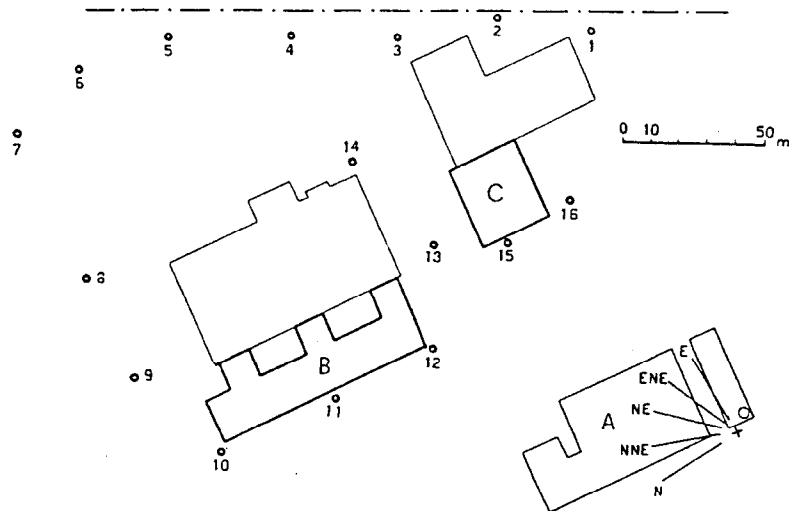
Figure 4: The 3-building complex downwind of the release location
marked as "+". The blocked spaces east of buildings B and C are
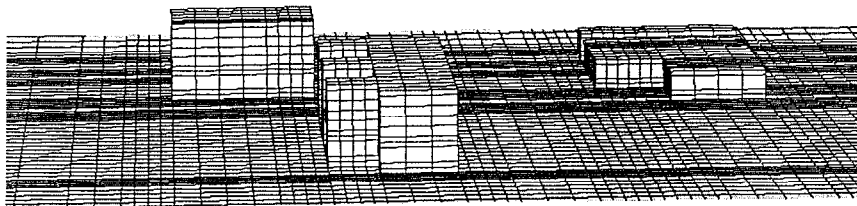parking areas. Also shown are the 16 surface sampling points.

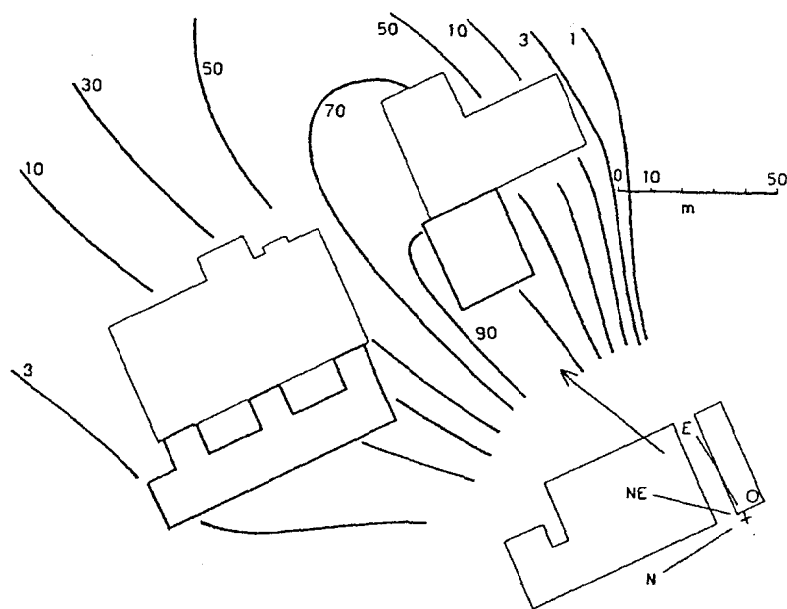Figure 5: The finite element mesh for the 3-building complex.

Figure 6: Concentration pattern for wind direction at 250°, 5.2 m/s at 10 m height. Units are given in 10^7 g/m^3.
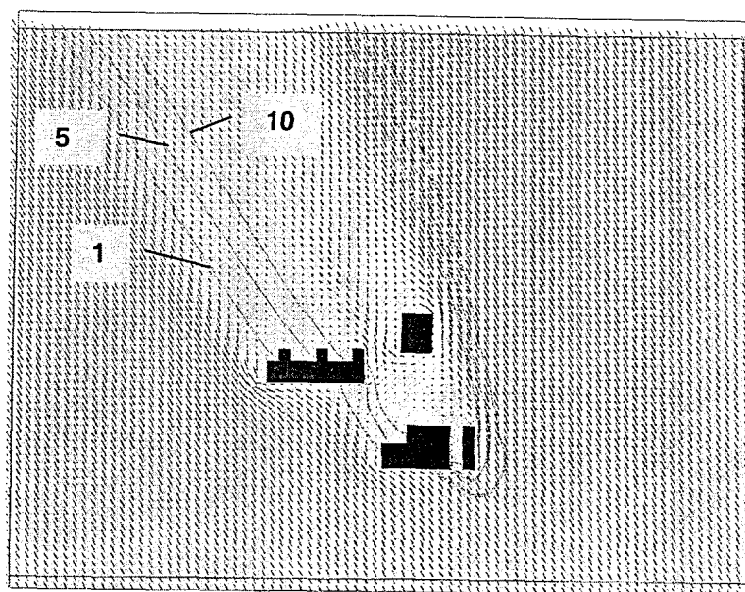

Figure 7: Simulated concentration pattern for the 250° wind field experiment. Units are given in 10^7 g/m^3.
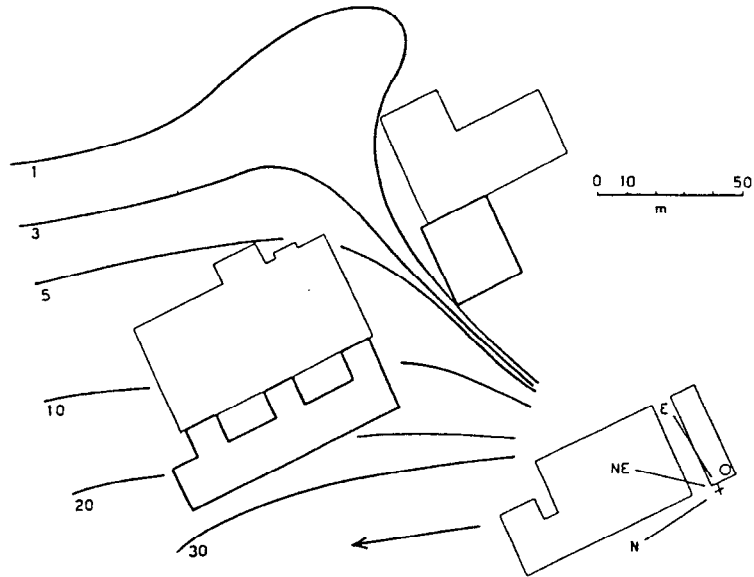
Figure 8: Concentration pattern for wind direction at 205°, 3.7 m/s at 10m height. Units are given in 10^7 g/m^3.
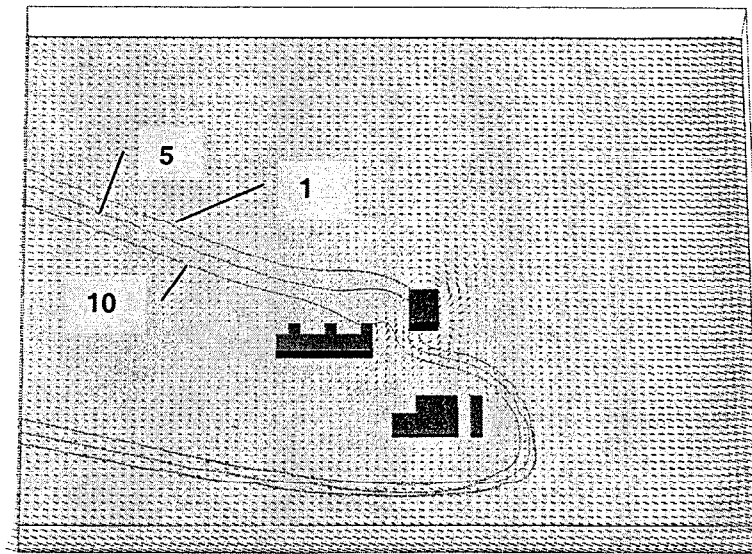

Figure 9: Simulated concentration pattern for 205° wind direction field experiment. Units are given in 10^7 g/m^3.